Evaluation of Tidal Currents Energetic Potential in Aveiro Harbor (Portugal) Using CFD Techniques

C. Rodrigues*[1], H. Calisto[2], N. Martins[3]

Department of Mechanical Engineering

University of Aveiro – Portugal

*[1]carlosfarinhas@ua.pt; [2]hcalisto@ua.pt; [3]nmartins@ua.pt

Abstract

Tidal currents are a renewable energy source,a study of their integration in existing energy nerworks is pertinent given their intrinsic sustainable nature.The potential of installing an array of electricity generators able to convert tidal current's kinetic energy into electricity at the entrance of Aveiro harbour was therefore evaluated. After analysing several technological options, a Verdant turbine was selected givenits adequate dimensions, suited tolocalconstraints. In order to simulate the tidal currents in the main canal of the Harbour, a hydrodynamic study was carried out by using a computational fluid dynamics (CFD) modelThe numerical simulations allow the identification of the most adequate places to implement the turbines according to their production capacity. Three different areas were identified and tagged with letters. An A class zone was found to be the one that could deliver more energy to the net. The B and C areas would respectively provide less energy. Experimental data wereused as a boundary conditions to characterize the water velocity along the canal.

Keywords

Tidal; Energy; CFD; Site Evaluation

Introduction

The gravitation field caused by the moon and the sun originates water movements in the oceans and consequently in estuary zones. This movement of water comprises an enormous energetic potential to be explored, Rourke et al. (2010) and Guney andKaygusuz (2010). Tranposing the concept of energyconversion from wind energy farms to water mass flows generated by the tidal cycles, a thorough study is carried out in orderto identify themost favourable places to implement similar technologies enablingthe exploitation of his source of renewable energy. The electric potential generated from sea currents is enormous, Guney (2011). The sea currents generated by the tide change have been being explored as a sustainable source of electric energy, Rourke*et al.* (2010). The forces generated by the currents and the fact that these are predictable days in advance make them particularly attractive to the generation of electric energy, with a comparative advantage to other renewable sources.The objective of this work consists in the evaluation of the electricity production potencial at the entrance of Aveiro Harbour using as primary energy source existing tidal currents. A literature survey was carried out regarding the available technologies to convert the tidal currents into electric powerwith the objective to examine, according to its dimensions and functionality, which was the technology that could be applied to the main navigation canal of Aveiro Harbour. Once the technology which best suits localconstraints was selected, a computational study was carried out regarding the preferential locations, in order to settle a farm of tidal turbines, having classified the canal regarding the distribution of the energetic potential as well as having verified the spacing between the turbines so as to optimize the park, maximizing the use of zones of greater potential and minimizing the negative interference, between the upstream and downstream turbines, originated by the wake phenomena.

The energy of the tidal currents has already been used a long time ago in Portugal. Tidal mills have been built since the 13th Century, being situated almost all over the country, from Minho to the Algarve, in estuaries and in river branches. An example of this use are the famous tidal mills in Almada, located in the estuary of the Tagus River, which taking advantage of the tidal cycles, grinded cereal.

At this moment, the technology used in producing electric energy from sea currents is still in an embryonic phase. The most consolidated solutions are

based on the technology of hydroelectric plants, adapted to the saline environment. However, the need for a dike limit the use of the estuary, namely regarding navigation, Khan et al. (2009) and Lago et al. (2010).

The oldest plant of this type was built in the estuary of the Rance River (France) in the 60's, with the power of 240 MW, Denny (2009). The current trend focuses on the use of open turbines (*free-flow*) in order to,Lago et al. (2010).*S. Bahaj, L. E. Meyers,*minimize the impact on the environment and the interference with other uses of the estuary and also the maintemance cost*(2003)* proposed models of one or two rotors with powers between 300 kW and 1 MW in which the electronic components are in a platform above the water level, facilitating the maintenance. However, the visual impact and navigability could be the negative aspect of this kind of solution. *BlueConcept* has already installed systems identical to horizontal axis wind turbines, with a power of 300 kW, see *Marine Current Turbines Limited (2014)*, and vertical axis wind turbines, installed on a floating platform, anchored by cables to the marine bottom, with a power of 20 kW, see e.g., *Blue Energy International Inc. (2014)* An identical solution is proposed by *Enermar*, with a direct connection between the turbine and the generator, i.e.., without gear boxes, see *Enermar System* (2007). *Rotech* developed a bi-directional 1 MW system, equipped with an external veturiduct,see *Lunar Energy Limited (2014)*.The venture duct increases the turbine efficiency as it improves the water velocity vector reaching the turbine blades, acting both in it intensity and direction. However, it larger diameter (due to the venture duct) may be a limitation in what concerns to some estuaries navigability. A very different system was developed by *Stingray*, in which the rotating machine is replaced by one of an alternative type, triggered by a hydroplane with an oscillating arm with a power of 150 kW, see *The Engineering Business (2014)*, presenting, nonetheless, limitations in navigation, due to the amplitude of the wing's movement. The solution which presented greater compatibility with the characteristics of the canal in study is proposed by *VerdantPower*, see *Technology Evaluation of Existing and Emerging Technologies (2006)*,consisting of *free-flow* type turbines, installed on top of submersed masts, set to the bottom of the canal, presenting characteristics in all aspects identical to the horizontal axis wind turbines, figure 1. Such turbines better known as KHPS, with a maximum power output of 35,9 KW, and it has a rotor of three blades with 5 meters of diameter, Bahaj and Meyers (2005).The pylon which supports the nacelle is 6 meters long.

FIGURE 1 – VERDANT POWER'S KHPS MODEL, FROMTECHNOLOGY EVALUATION OF EXISTING AND EMERGING TECHNOLOGIES (2006)

The nacelle as components like an epicyclical multiplier guided by the rotor, which, in turn, sets into motion a three phase induction motor which provides electrical energy to the network. The material which covers the nacelle is of steel covered by fibre glass and *epoxy,* allowing high resistance and being highly impermeable.The power curve is presented in figure 2, revealing a maximum capacity of 38 kW obtained for a current velocity of around 2.2 m/s, requiring a minimum of 1 m/s to start.

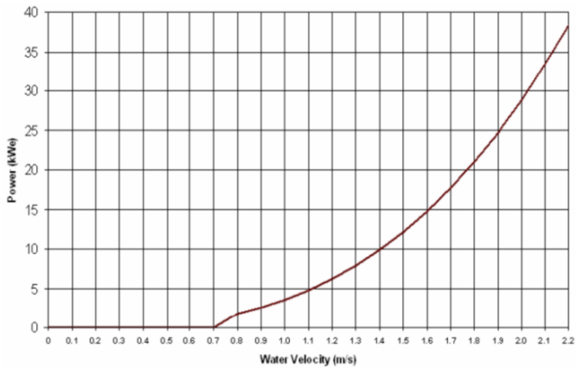

FIGURE 2 – POWER CURVE OF THE KHPS TURBINE,SEE TECHNOLOGY EVALUATION OF EXISTING AND EMERGING TECHNOLOGIES (2006)

TABLE 1 - CHARACTERISTICS OF THE KHPS TURBINE

Characteristics	Minimum	Maximum	Units
Velocity	1.0	2.1	m/s
Capacity	4.2	35.9	KW
Depth of the canal	6	20	M
Length of the canal	6	-	M
Efficiency of the turbine	38.5		%

The main characteristics which support the compatibility of this option with the type of use regarding the canal under study are summarized in table 1, namely regarding the rotor diameter, the

depth of the canal and the gauge of the vessels received.

The turbines are completely submerged and their maintenance is performed below the surface. While being a possible inconvenient, this has the advantage of avoiding the visual evidence of the existence of the farm

Simulation of the Flow in the Canal

The assessement of the available tidal energy resources in a certain location is a current research topic. Different approaches and models are used by reserchers, usually mixing experimental data and numerical predictions from computational models, Grabbe*et al.* (2009). In the present work, experimental speed velocity data provided by the Department of Civil Engineering of the University of Aveiro (DECUA) were used, making possible thedrawing of tables related to the different phases of the tides during the period of spring-tides and neap-tides in Winter and in Summer. The chart regarding the velocity profile presented in figure 3 was used for the simulation of a high tide in the navigation canal and defined as an inlet condition.

For the simulation, CFX® 5.6 was used in order to determine the zones which are more favourable in terms of speed in order to place the turbines and allow a better visualization of the wake caused by them in the navigation canal. The experimental characterization of the currents was carried out with an ADCP (*Acoustic Doppler Current Profiler*), being used to measure the current velocity of the water at different depths.

The velocity profiles above representedwere measured in a section of the canal which linked the zone of the tide graph up to the north pier of S. Jacinto.

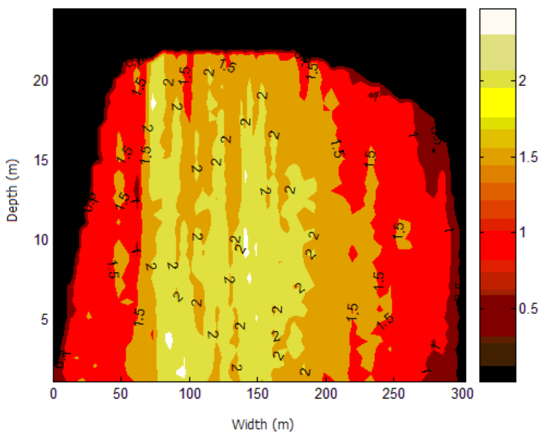

FIGURE 3 – VELOCITY (M/S) OF THE 1ST HIGH TIDE ON 28/02/02

An approximate value of 4866 m² was calculated for the area of this section. In the same place, a project developed by researchers from the Department of Physics from the University of Aveiro (DFUA), was taking place. Within the framework ofthat project volume flow rates were measured during a successive period of two months with time intervals of six minutes. For the section previously mentioned it was possible to calculate the current velocity regarding each instant, figure 4. The period in question refers to the time interval from September 3rd (Julian day 243) to November 13th (Julian day 317), 2002.

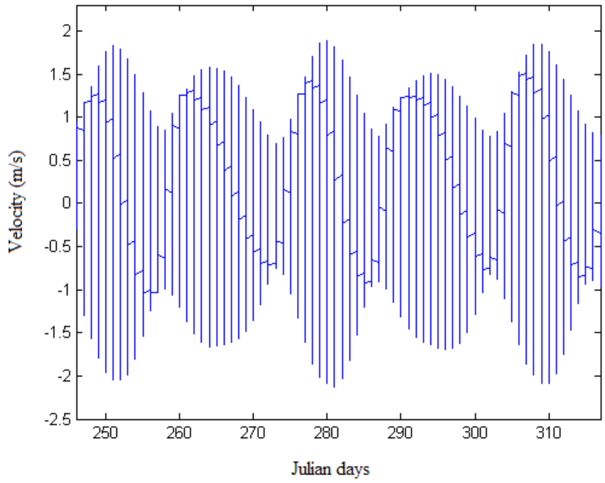

FIGURE 4 – VELOCITY OF THE CURRENT BETWEEN 03/09/02 AND 13/11/02

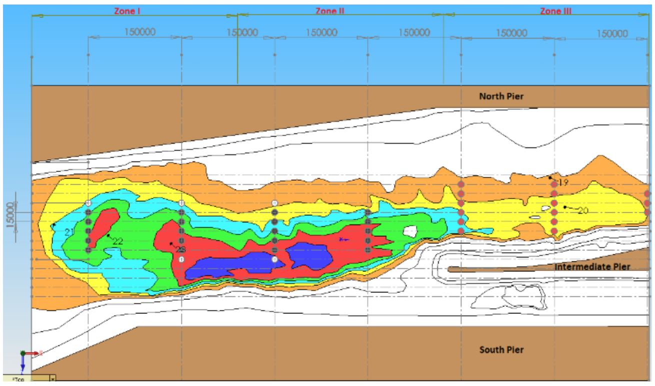

FIGURE 5 – CALCULATION DOMAIN DIVIDED INTO THREE ZONES.

An analysis and simulation of the flow in the main navigation canal of Aveiro Harbour, figure 5, was carried outtogether with the evaluation of its respectiveenergetic potential, adopting the methodology represented in figure 7.

For the outlet section of the calculation domain, which corresponds to the section between the intermediate pier and the north pier of São Jacinto (Zone III), represented in figure 5 and 12, the zero pressure

gradient boundary condition was used, where the parameter to be defined is the relative pressure of the water in the normal direction to the section, that is, in the direction of the flow.

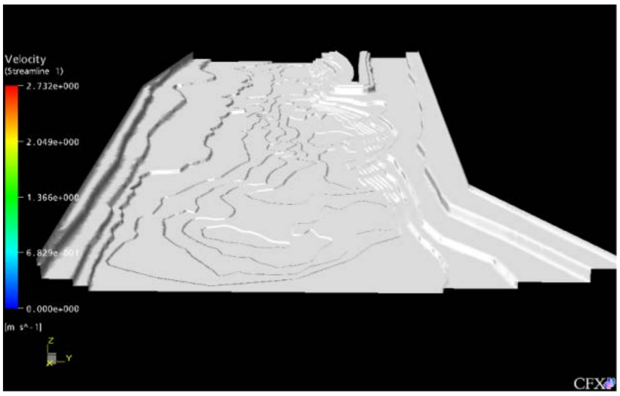

FIGURE 6–MAIN NAVIGATION CANAL OF THE AVEIRO HARBOUR

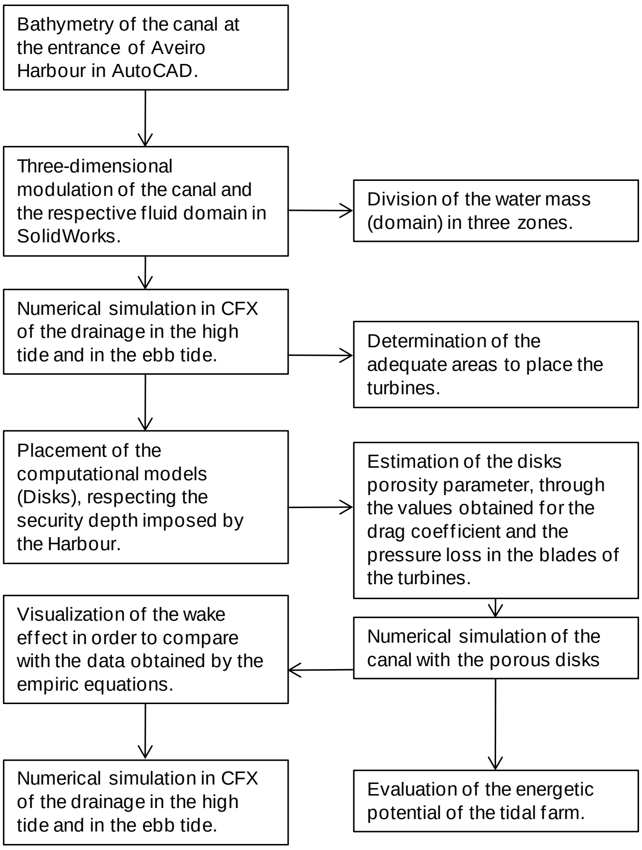

FIGURE 7– ADOPTED METHODOLOGY

In the entrance section of the calculation domain an imposed velocity boundary condition was used. The velocity values used were the experimental values given by the Department of Physics from the University of Aveiro (DFUA). In the interface of the liquid and the gaseous phase, which corresponds to the surface of the water, a symmetry plane boundary condition was used, which reflects the inexistence of shear stresses generated by the air in the water's surface. An approach carried out was to consider that there is no normal velocity on the surface of the liquid phase; meaning that for the study regarding the fluid behaviour along the canal, the vertical movement of the water mass in the interface water-air is neglected. A fluid-solid no-slip boundary condition was specified on the lower boundary of the domain, which corresponds to thebottom of the canal. These conditions were, in general, applied to all simulated domains of water masses. The standard k-ε turbulenceclosure modelwas used. The existence of aquatic vegetation or the movement of the sedimentary layers was not taken into account.

Methodology Used to Estimate the Annual Energy Produced.

It should be noted that to estimate the annual energy produced, the data supplied by DFUA wereused. Whereas for the numerical simulation, data supplied by DECUA was used, regarding a period of spring-tide. Equation 1 corresponds to the power curve of the turbine as a function of the water velocity, graphically represented in figure 2, available at the report. Technology Evaluation of Existing and Emerging Technologies (2006)

$$P_{curve}(KW) = 16.7 \times v^{2} - 25 \times v + 13.3 \qquad (1)$$

The energy provided by a turbine at the end of a month was calculated as follows: the peaks of power during the period from September 3rd to November 13th were added and it was multiplied by the interval of the measurements.

$$E(KWh) = \sum_{day246}^{day317} P_{curve}(i) \times \Delta T \qquad (2)$$

The average energy that each turbine is able to provide at the end of the month is approximately 2847 kWh for velocities greater than 1 m/s. After one year, each turbine would be able to provide the network about 34200 kWh. The class B turbines will occupy the largest area of the canal, which in turn comprehends the measurement area of volume flow rates during the period from September 3rd to November 13th of 2002, for flow rate measurement intervals of 6 minutes.

As a result of the flow simulation along the canal, contour charts were drawn for average depths of 13 meters (rotor axis) with the objective to determine the flow velocities calculated by means of computational software,figures 8and9.

After the analysis of the contour charts, it was possible

to determine the maximum velocities for each area (A,B and C) and it was possible to estimate the following correction factors during the high tide and the ebb-tide:

$$1.13 \times V_{BE} = V_{AE} = \frac{V_{CE}}{0.87} \tag{3}$$

$$1.07 \times V_{BV} = V_{AV} = \frac{V_{CV}}{0.97} \tag{4}$$

These correction factors were used in order to estimate the turbines power output ate the locations A, B and C.

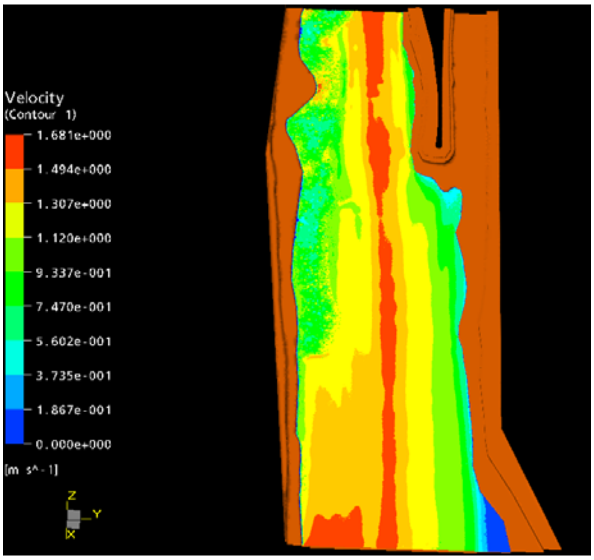

FIGURE 8–VELOCITY CONTOURS AT A DEPTH OF 13 M (HIGH TIDE)

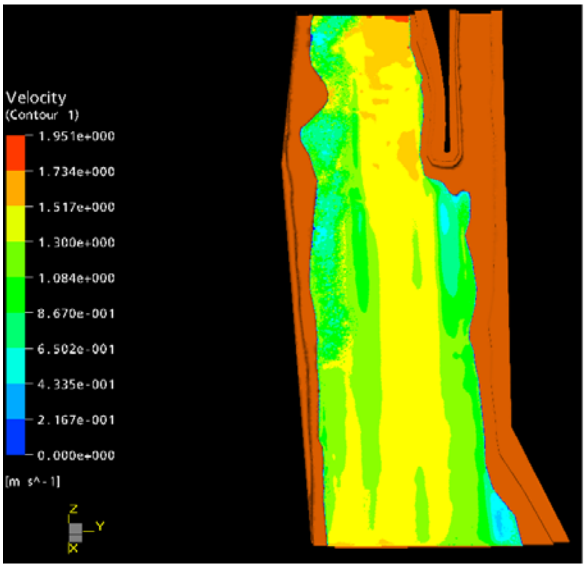

FIGURE 9– VELOCITY CONTOURS AT A DEPTH OF 13 M (EBB-TIDE)

Assuming that the respective velocity ratios are kept constant throughout the several tidal cycles, it is possible to draw the velocity charts, (figures 10and 11) over the period from September 3rd to November 13th.

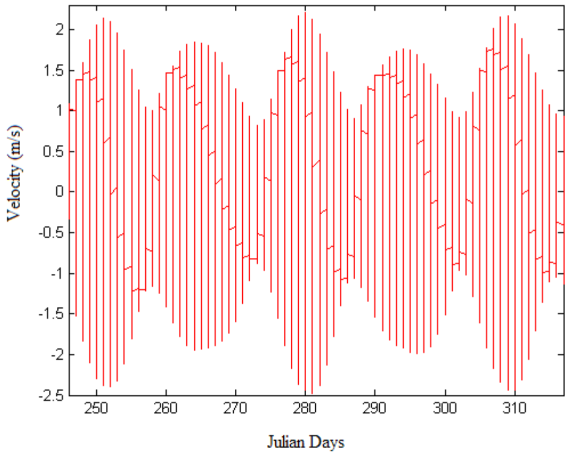

FIGURE 10– VELOCITY BETWEEN 2/09/02 AND 13/11/02 FOR CLASS A TURBINES.

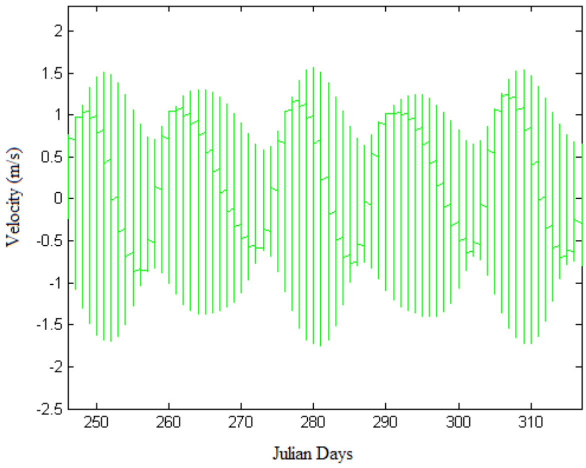

FIGURE 11– VELOCITY BETWEEN 2/09/02 AND 13/11/02 FOR CLASS C TURBINES.

Figures 4, 10and 11correspond to average velocity values of the flow obtained from the value of the volume flow rate measured every 6 minutes, in the section of the canal which links the area of the tide graph to the north pier of S. Jacinto. Using the velocity data obtained with the numerical simulations, (figures 10and 11) and equations (1), (2), (3) and (4) to calculate the energy provided by the turbines, it was possible to conclude that each class A and C turbine is able to provide the network with about 47300 kWh and 23400 kWh during one year respectively.

Verification of Empiric Methods

The empiric rules of thumb used to establish the distances between turbinestake in account the wake caused by the drag force when the water passes through the blades. The longitudinal and transversal distances between the turbines are given by the following equations:

$$D_L = \alpha \times D \tag{5}$$

$$D_T = \beta \times D \tag{6}$$

While α is the parameter, which determines the lateral spacing between the turbines, along a section of the canal, β is the parameter which determines the spacing between the turbines along the length of the canal. The empirical values for α and βmay be taken as 3 and 30 respectively, for details see *The exploitation of tidal and marine currents*, Report EUR 16683 EN.

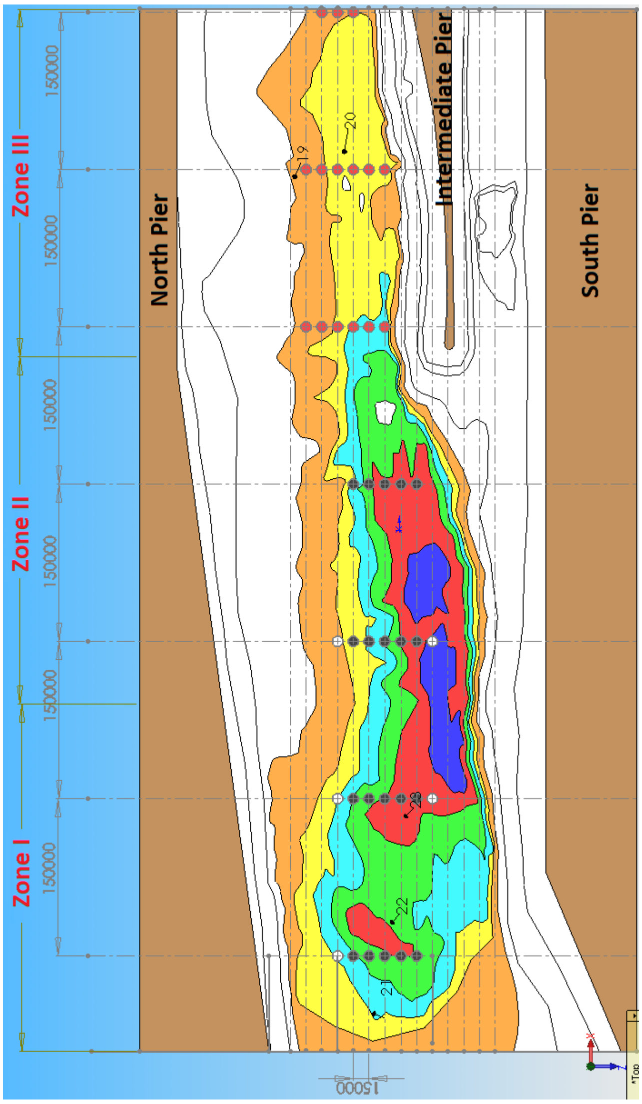

FIGURE 12– LAYOUT OF THE TIDAL FARM

Given the dimensions of the canal to simulate, it was necessary to have in mind the complexity of the geometry and the high computational cost. As an example, a mesh for a complete simulation of the canal, without the turbine model, contains approximately 2,8 million elements (tetrahedra). The creation of a mesh of the global canal with the models of the turbines in it was not possible due to the fact that the availablecomputationalresources were not able to process the amount of information. The first step to overcome such limitation was the subdivision of the overall domain (water mass) into three parts, designated as Zone I (water mass zone closer to the sea), Zone III (water mass zone between the north pier and the intermediate pier) and zone II (intermediate water mass zone), figure 12.

The turbines represented in red symbolize the ones of class A (zone III), and the class B and C ones are represented in black and white, respectively (zone I and II). The turbines are distanced from each other 150 meters along the canal's lenght and 15 meters along itswidth.

Modelling KHPS Turbines Using Darcy's Approximation

In order to test whether it would be possible to use approximate models of the turbines in the computational domain, fractions of water mass were used corresponding to three zones, figure 12. The computational model of the turbine was introducedin these domains. During the generation of the mesh, several errors occurred due to the impossibility of refining the mesh near the surfaces of the model of the turbine due to the dimension of it being too small compared to the dimensions of the canal. In a second approach, a less complex turbine geometry was elaborated. The real turbine model was modelled by an approximated geometry, where the support pillar was a continuation of the nacelle, ending in a disk which intended to simulate the blades of the turbines, figure 13.

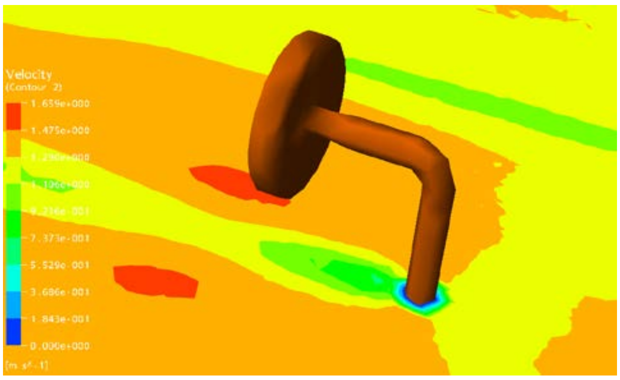

FIGURE 13– TURBINE MODEL CONSIDERED IN CFD SIMULATIONS.

It was verified, through a numerical simulation, that the wake caused by the pillar is negligible when compared to the wake caused by the disk.Thus, following the procedure proposed by M.E. Harrison *et.*

al (2009), the turbines were modeled as if they were porous disks with the thickness of approximately the same as the thickness of the blade, which could offer the same resistance to the passage of the fluid,. For the effect, the *Darcy* equation, (7) was used, which describes the pressure loss when a fluid crosses a certain region.

$$-\frac{\partial P}{\partial x_i} = \frac{\mu}{K} \times u_i + K_{loss} \times \rho \times |u| \times u_i \tag{7}$$

The methodology used was:

1. Calculation of the approximate value of the real pressure loss when the fluid passes through the turbine blades.
2. Introduction of the value in the *Darcy* equation, and determination of the porosity parameter, knowing the pressure loss coefficient for porous environments, Idelchik, I.E (1994).

Through literature survey regarding the dimensioning of *NACA* profiles, two original equations from the *general flat plate theory*, S. Bahaj, L. E. Meyers. (2005) were obtained:

$$C_D = 1.28 \times \sin\alpha \tag{8}$$

$$C_L = 2 \times \pi \times \alpha \tag{9}$$

Given the impossibility of obtaining the blades´angle of attack for the considered technology, due to it confidential nature, it was decided to consider 30º sinceitis in the interval considered by other authors in other studies envolving the same type of blades, S. Bahaj, L. E. Meyers. (2005). Thus a value of 0.64 was obtained for the drag coefficient (8). For the loss coefficient, the value of 0.7 was calculated, Idelchik, I.E (1994)..

$$\Delta P = \frac{F_D}{A_F} = C_D \times \frac{1}{2} \times \rho \times u^2 \tag{10}$$

A variant of the *Darcy* equation(11), , was used for the 1st point of the methodology adopted to address computational models of porous disks as if these represented the blades of a tidal turbine,for a detailed derivation see ANSYS CFX – Solver Theory Guide. After having calculated in (10) the range of approximate values for the pressure loss difference in the preferential zones (greater current velocity), the resistance parameters present in the *Darcy* equation were calculated, which can be introduced in the boundary condition *(porous media)* applied to the sub domain (Disks). According to the 2nd point of the methodology, the known parameters and the differential pressure obtained in the previous steps are introduced now, using a variant of the *Darcy* equation:

$$\Delta P = \underbrace{\frac{\mu}{K \times \beta}}_{C^{R1}} \times u \times l + \underbrace{\frac{K_{Loss} \times \rho}{2 \times \beta^2}}_{C^{R2}} \times u^2 \times l \tag{11}$$

In the equation described above in (11), the linear resistance term which varies linearly with the speed will not be taken into consideration, and only the quadratic resistance term which varies with the square speed will be taken in account (most important term) because plausible values for the permeability parameter were not found in literature. Parameter β originates values between 1,2 and 1,4. The quadratic resistance term which is introduced as a boundary condition of the sub domain (disks), in the software was found to be between 368 and 501 [$Kg.m^{-4}$].

Through such data, the same methodology was adopted for the several zones chosen for the implementation of the turbines.

Analysis of Results

As can be seen in configuration A (the turbines aligned upstream and downstream), represented by figure 14, the empirical distances are respected and the wake does not influence the turbines immediately upstream. The fraction of the canal represented, fig. 15, refers to the zone where the velocities reach greater speed, between the intermediate pier and the north pier of S. Jacinto.

The disks, which correspond to the turbines, were placed at 15 meters away from each other along the width of the canal, and at 150 metres along its length (empirical parameters).

It was verified that the distance originated by the wake is respected in the first 4 turbines.

In the turbine close to the intermediate pier, we verify that its wake affects the turbines lined up upstream which in turn will not be as efficient.

Therefore, it was observed that this configuration (B) with the turbines out of line and respecting the empirical distances is not the most adequate, but it shows improvements comparing to the initial configuration (A), for the same fraction of the canal, in which they were aligned.

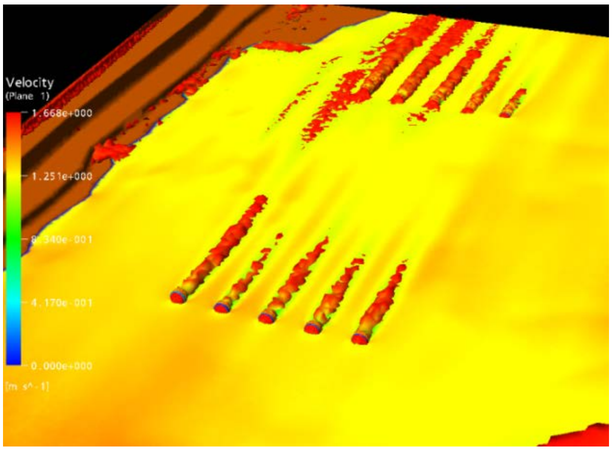

FIGURE 14– WAKE CREATED BY THE RESISTANCE OF THE DISKS TO THE FLUID DURING THE HIGH TIDE IN THE FIRST PART OF THE CANAL (CONF. A), FOR VELOCITY VALUES BETWEEN 0 AND 1,4 M/S.

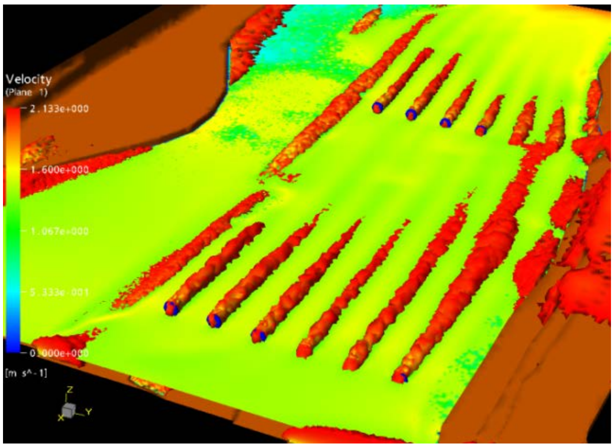

FIGURE 15– WAKE CREATED BY THE RESISTANCE OF THE DISKS TO THE FLUID DURING A HIGH TIDE IN THE 3RD PART OF THE CANAL (CONF. B), FOR VELOCITY VALUES UP TO 1,35 M/S.

Conclusions

In this work, a study of a set of appropriate technologies for the exploitation of tidal current energy was described, presenting their main characteristics and potentialities. Among the technologies analysed, a tidal turbine from Verdant was selected given its compatibility with the entrance of Aveiro Harbour. Through the application of the CFX® 5.6 computational fluid dynamics software, a hydrodynamic study of the navigation canal was carried out to verify the possible configuration of a tidal farm. After having performed a simulation for configuration A (aligned turbines upstream and downstream), a less efficient zone in terms of energy was identified next to the intermediate pier in the third part of the canal, where the wake caused by two turbines affects those aligned downstream. This phenomenon can also be due to the particular geometry of the canal and the proximity of the turbines to the intermediate pier. An alternative configuration (B, figure 14) where the turbines were out of line respecting the same empirical distances shows an improvement, verifying that only one of the piers affected the turbines immediately downstream.

The analysis developed in this study demonstrates the potential and importance of the use of advanced flow simulation tools in engineering projects of this nature. The basic methodology leading to its conclusions, in terms both of geometry and boundary condition processing prior to the simulations and of analysis, is possible to apply to similar situations at other locations, enabling a justification of the adequate way to locate the turbines within the available space, as well as allowing an estimate of penalty factors associated with different possible locations.

ACKNOWLEDGEMENTS

We would like to thank Dr.ª Rita Nolasco of the Department of Physics and Prof. Dr. Carlos Coelho of the Department of Civil Engineering from the University of Aveiro for providing data, namely speeds and flows in the canal and also Dr. Sérgio Dias from the Administration of Aveiro Harbour, for the availability and information provided regarding the canal and its use.

REFERENCES

Ansys CFX – Solver, Theory Guide, Ansys CFX Release 11.0, December 2006.

A.S. Bahaj, L. E. Meyers. (2003); Fundamentals applicable to the utilisations of marine current turbines for energy production.*Renewable Energy 28,* 2205-2211

A.S. Bahaj, L. E. Meyers. (2005); Power output performance characteristics of horizontal axis marine current turbine. *Renewable Energy 31,* 197 – 208

Blue Energy International Inc. homepage, visited in 30/03/2014; **www.bluenergy.com**

Enermar System, CA-OE Workshop, April 25 Copenhagen (2007)(http://www.spok.dk/seminar/Enermar%20system _32.pdf)

E. Denny (2009); The economicsoftidalenergy; Energy Policy, 37, 1914–1924; doi:10.1016/j.enpol.2009.01.009

F. O.Rourke, F. Boyle, A. Reynolds (2010); Tidal energy update 2009; Applied Energy, 87, 398–409; doi:10.1016/j.apenergy. 2009.08.014

Idelchik, I.E., *Handbook of Hydraulic Resistance*, CRC Begell House, 1994

L.I. Lago, F.L. Ponta, L. Chen (2010); Advances and trends in hydrokinetic turbine systems.Energy for Sustainable Development 14, 287–296;doi:10.1016/j.esd.2010.09.004

Lunar Energy Limited homepage in 30/03/2014; **www.lunarenergy.co.uk**

Marine Current Turbines Limited homepagevisitedin 30/03/2014; **www.marineturbines.com**

M. Grabbe, E.Lalander, S.Lundin, M. Leijon (2009);A review of the tidal current energy resource in Norway; Renewable and Sustainable Energy Reviews, 13, 1898–1909;doi:10.1016/j.rser.2009.01.026

M. J. Khan, G. Bhuyan, M.T. Iqbal, J.E. Quaicoe, (2009); Hydrokinetic energy conversion systems and assessment of horizontal and verticalaxis turbines for river and tidal applications: A technology status review; Applied Energy, 86, 1823–1835;doi:10.1016/j.apenergy.2009.02.017

M.S. Guney, K. Kaygusuz (2010); Hydrokinetic energy conversion systems: A technology status review; Renewable and Sustainable Energy Reviews, 14, 2996–3004;doi:10.1016/j.rser.2010.06.016

M. S.Guney (2011); Evaluation and measures to increase performancecoefficient of hydrokinetic turbines; Renewable and Sustainable Energy Reviews, 15, 3669–3675; doi:10.1016/j.rser.2011.07.009

M.E. Harrison, W.M.J. Batten, L. E. Meyers, A.S. Bahaj; (2009); A comparison between CFD simulations and experiments forpredicting the far wake of horizontal axis tidal turbines. Proceedings of the 8th European Wave and Tidal EnergyConference, Uppsala, Sweden, 2009

The Engineering Business. (2003); Stingray Tidal Stream Energy Device – Phase 2; Manual, 16-30; (**www.engb.com**)

Technology Evaluation of Existing and Emerging Technologies - Water Current Turbines for River Applications (2006), Prepared by Verdant Power Canada ULC for Natural Resources Canada under Contract NRCan-06-01071, June 15,

The exploitation of tidal and marine currents, Report EUR 16683 EN, DG Science, Research and Development, ISBN 92-827-5658-0; 1996

Carlos Rodrigues Mechanical Engineer at Universidade de Aveiro in July 2007. MSc in Mechanical Engineering at Universidade de Aveiro, in December 2008. Post-graduation course in renewable energies at TÜV Akademie, in July 2011. Works at Firstep, developing new products based on renewable energies.

Hugo Calisto PhD in Mechanical Engineering in July 2013 with Thesis “Transient Transpiration Radiometer – Development of a Heat Flux Sensor for High Aggressivity Environments”.

He was Lecturer at Universidade de Aveiro between February 2003 and September 2013., Consultant on several Energy Efficiency-related programmes.

Dr. Calisto is Scientific Officer at the Energy Security Unit - Institute for Energy and Transport, Joint Research Centre, European Commission, since September 2013.

Nelson Martins completed his PhD at Instituto Superior Técnico in 1999 with the thesis "The new heat flux meter for high temperature harsh environments." In recent years he has been responsible for coordinating the field of Energy at the Department of Mechanical Engineering, University of Aveiro, His research activity is focused in the sustainable energy systemsarea, being divided on issues such as monitoring and diagnostic processes, nanofluidsdevelopment for energy systems, analysis of energy systems in industry and buildings.

As an engineer in the AVAC and Energy Efficiency sector he is a Senior member of the Portuguese Energy Society, a Qualified Expert of the Portuguese System for Energy Certification of Buildingsansd In the Indstry and a CertifiedExpert of the InternationalPerformance Measurement and VerificationProtocolby the Efficiency Valuation Organization (EVO) and the Association of Energy Engineers (AEE).

Dr. Martins currently is Assistant Professor in the Department of Mechanical Engineering, University of Aveiro. He